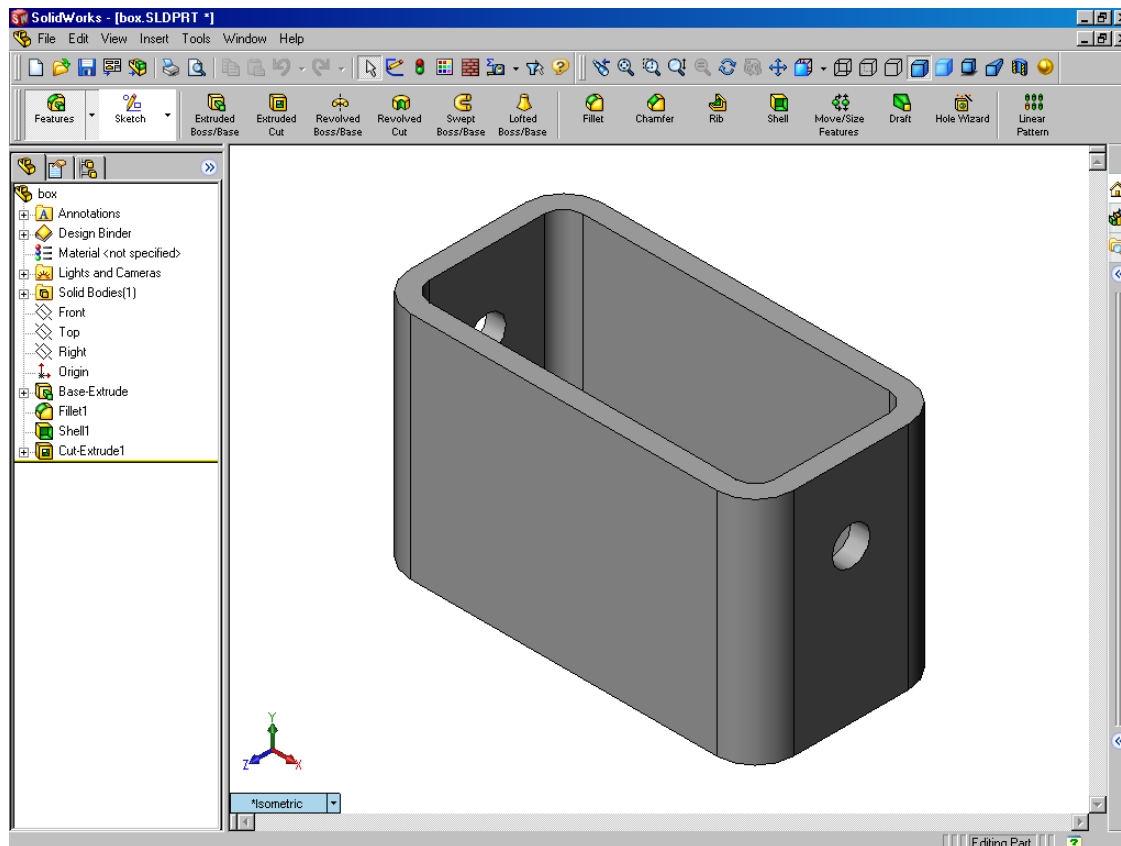


Lesson 2: Basic Functionality

Goals of This Lesson

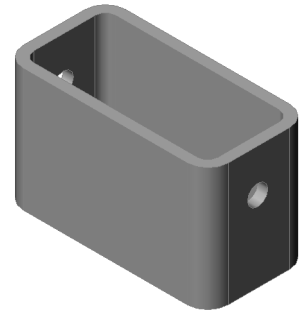
- Upon successful completion of this lesson, you will be able to understand the basic functionality of SolidWorks software and create the following part:




Active Learning Exercise — Creating a Basic Part

Use SolidWorks to create the box shown at the right.

The step-by-step instructions are given below.



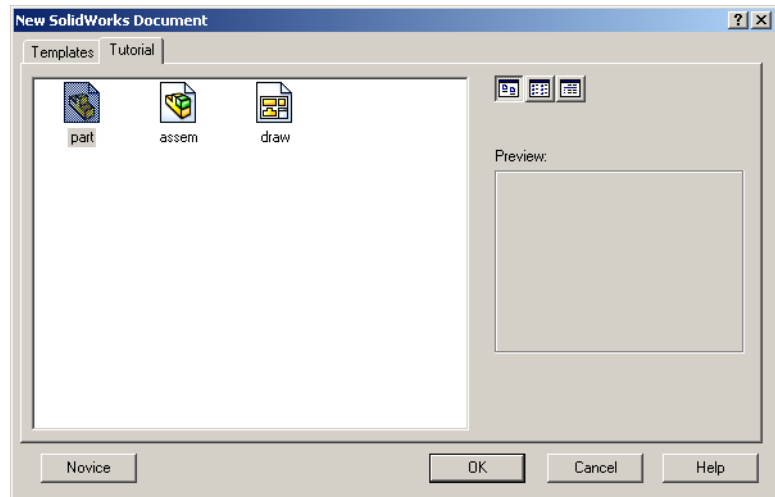
Create a New Part Document

- 1 Create a new part. Click  on the Standard toolbar.

The **New SolidWorks Document** dialog box appears.

- 2 Click the **Tutorial** tab.
- 3 Select the **Part** icon.
- 4 Click **OK**.

A new part document window appears.



Base Feature

The Base feature requires:

- ☐ Sketch plane – **Front** (default plane)
- ☐ Sketch profile – 2D Rectangle
- ☐ Feature type – Extruded boss feature

Open a Sketch

- 1 Open a 2D sketch. Click  on the Sketch toolbar.
Move the pointer over the **Front** plane, and click to select it.

Confirmation Corner

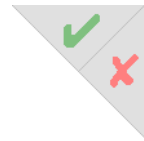
When many SolidWorks commands are active, a symbol or a set of symbols appears in the upper right corner of the graphics area. This area is called the **Confirmation Corner**.

Sketch Indicator

When a sketch is active, or open, a symbol appears in the confirmation corner that looks like the **Sketch** tool. It provides a visual reminder that you are active in a sketch. Clicking this symbol exits the sketch saving your changes. Clicking the red X exits the sketch discarding your changes.

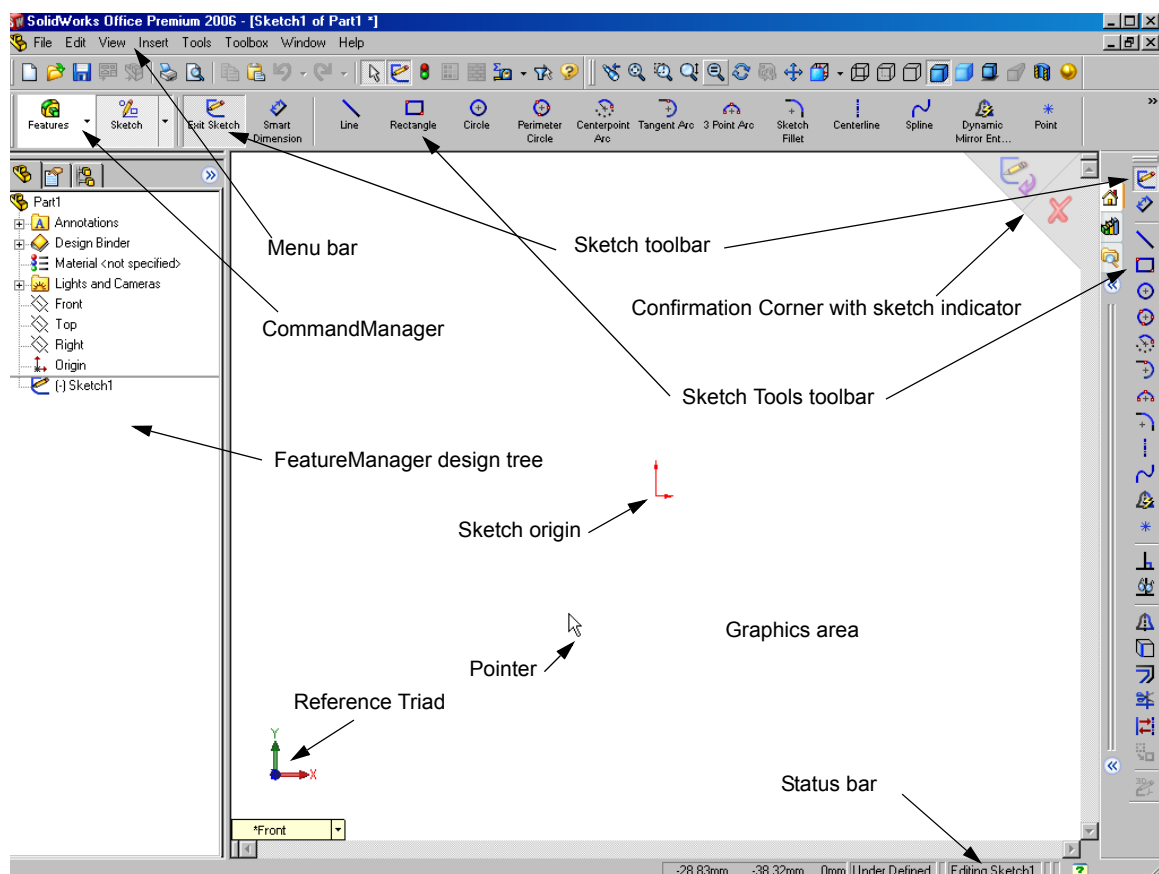


When other commands are active, the confirmation corner displays two symbols: a check mark and an X. The check mark executes the current command. The X cancels the command.




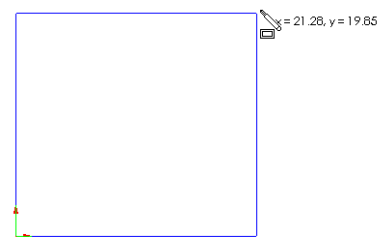
Overview of the SolidWorks Window

- ❑ A sketch origin appears in the center of the graphics area.
- ❑ The Sketch Tools and Sketch Relations toolbars are displayed.
- ❑ “Editing Sketch” appears in the status bar at the bottom of the screen.
- ❑ Sketch1 appears in the FeatureManager design tree.
- ❑ The status bar shows the position of the pointer, or sketch tool, in relation to the sketch origin.






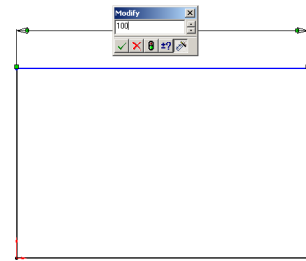
Sketch a Rectangle


- 1 Click  on the Sketch Tools toolbar.
- 2 Click the sketch origin to start the rectangle.
- 3 Move the pointer up and to the right, to create a rectangle.
- 4 Click the mouse button again to complete the rectangle.

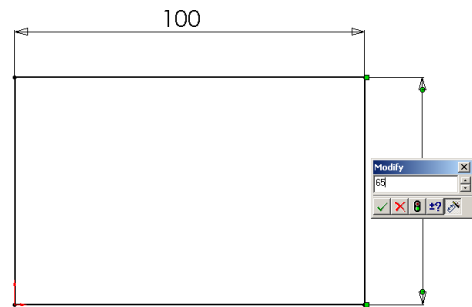


Add Dimensions

- 1 Click **Smart Dimension**  on the Sketch toolbar.
The pointer shape changes to .
- 2 Click the top line of the rectangle.
- 3 Click the dimension text location above the top line.
The **Modify** dialog box is displayed.
- 4 Enter **100**. Click  or press **Enter**.





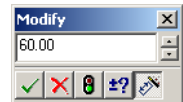
- 5 Click the right edge of the rectangle.
 - 6 Click the dimension text location. Enter **65**.
Click .
- The top segment and the remaining vertices are displayed in black. The status bar in the lower-right corner of the window indicates that the sketch is fully defined.



Changing the Dimension Values

The new dimensions for the box are 100mm x 60mm. Change the dimensions. Use the **Select** tool.

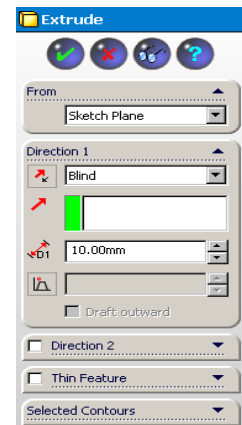
- 1 Click **Select**  on the Standard toolbar.
- 2 Double-click **65**.
The **Modify** dialog box appears.
- 3 Enter **60** in the **Modify** dialog box.
- 4 Click .



Extrude the Base Feature.


The first feature in any part is called the *Base Feature*. In this exercise, the base feature is created by extruding the sketched rectangle.

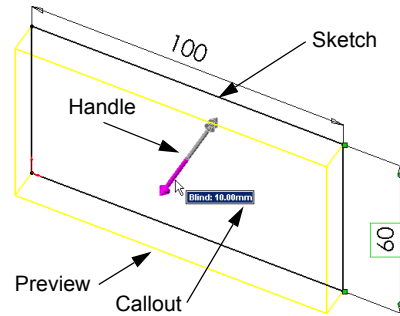
- 1 Click **Extruded Boss/Base**  on the Features toolbar.
The **Extrude Feature** PropertyManager appears. The view of the sketch changes to isometric.




2 Preview graphics.

A preview of the feature is shown at the default depth.

Handles  appear that can be used to drag the preview to the desired depth. The handles are colored magenta for the active direction and gray for inactive direction. A callout shows the current depth value.




Click **Detailed Preview**  in the Property Manager to see the feature in shaded preview mode.

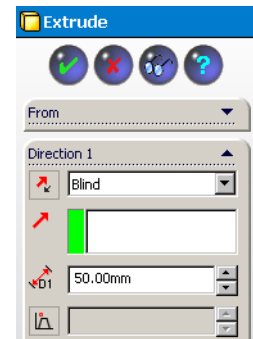
The cursor changes to . If you want to create the feature now, click the right mouse button.

Otherwise, you can make additional changes to the settings. For example, the depth of extrusion can be changed by dragging the dynamic handle with the mouse or by setting a value in the PropertyManager.

3 Extrude feature settings.

Change the settings as shown.

- End Condition = **Blind**
-  (Depth) = **50**



4 Create the extrusion. Click **OK** .

The new feature, `Extrude1`, is displayed in the FeatureManager design tree.

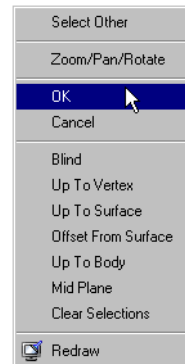
TIP:


The **OK** button  on the PropertyManager is just one way to complete the command.

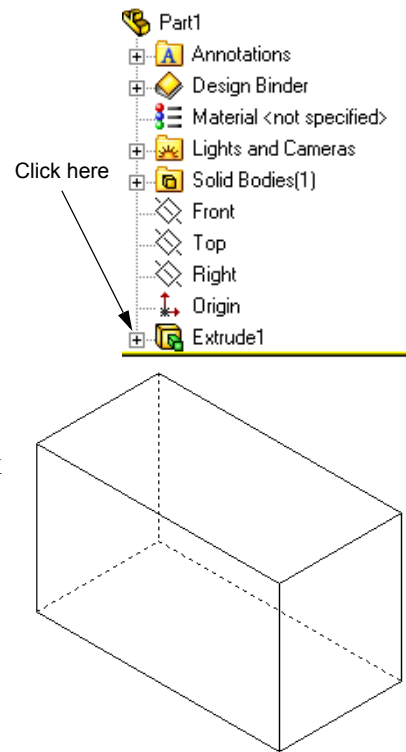
A second method is the set of **OK/Cancel** buttons in the confirmation corner of the graphics area.




A third method is the right-mouse shortcut menu that includes **OK**, among other options.



- Click the plus sign  beside **Extrude1** in the FeatureManager design tree. Notice that **Sketch1** — which you used to extrude the feature — is now listed under the feature.




View Display

Change the display mode. Click **Hidden Lines Visible**  on the View toolbar.

Hidden Lines Visible allows you to select hidden back edges of the box.

Save the Part

- Click **Save**  on the Standard toolbar, or click **File, Save**.

The **Save As** dialog box appears.


- Type **box** for the filename. Click **Save**.

The **.sldprt** extension is added to the filename.

The file is saved to the current directory. You can use the Windows browse button to change to a different directory.

Round the Corners of the Part

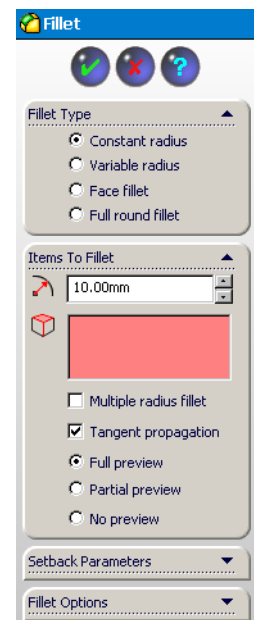
Round the four corner edges of the **box**. All rounds have the same radius (10mm). Create them as a single feature.

- Click **Fillet**  on the Features toolbar.

The **Fillet** PropertyManager appears.

- Enter **10** for the **Radius**.

Leave the remaining settings at their default values.






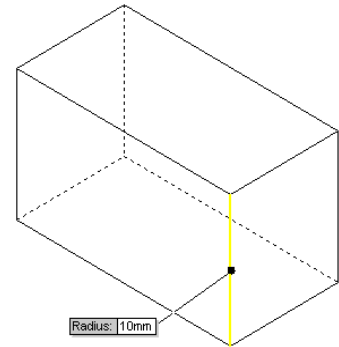
- 3 Click the first corner edge.

The faces, edges, and vertices are highlighted as you move the pointer over them.

When you select the edge, a callout **Radius: 10mm** appears.

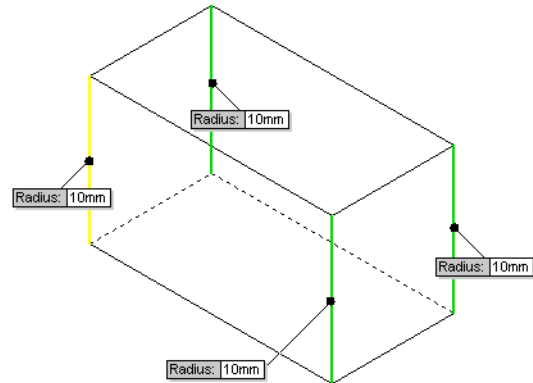
- 4 Identify selectable objects. Notice how the pointer changes shapes:

Edge:  || Face:  Vertex:  □



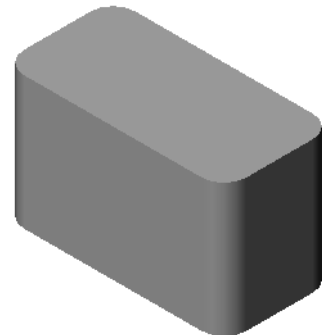
- 5 Click the second, third and fourth corner edges.

Note: Normally, a callout only appears on the *first* edge you select. This illustration has been modified to show callouts on each of the four selected edges. This was done simply to better illustrate which edges you are supposed to select.




- 6 Click **OK** .

Fillet1 appears in the FeatureManager design tree.



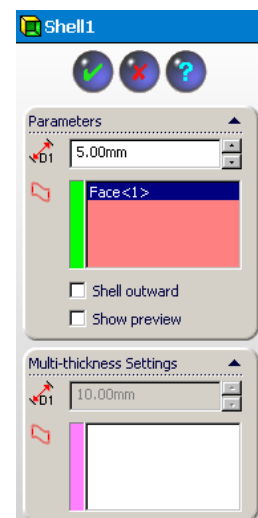
Hollow Out the Part

Remove the top face using the Shell feature.

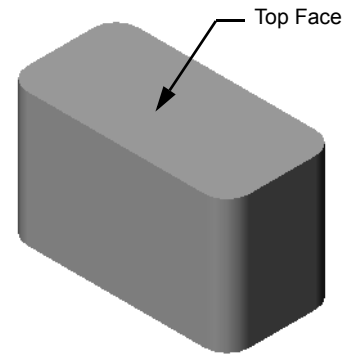
- 1 Click  on the Features toolbar.

The **Shell Feature** PropertyManager appears.

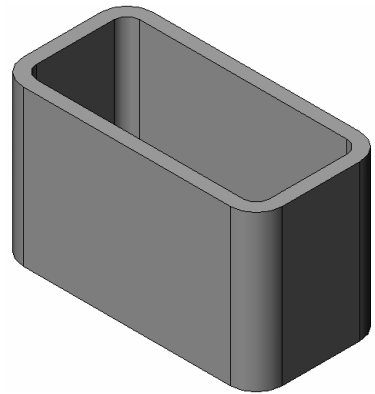
- 2 Enter **5** for **Thickness**.



- 3 Click the top face.



- 4 Click .





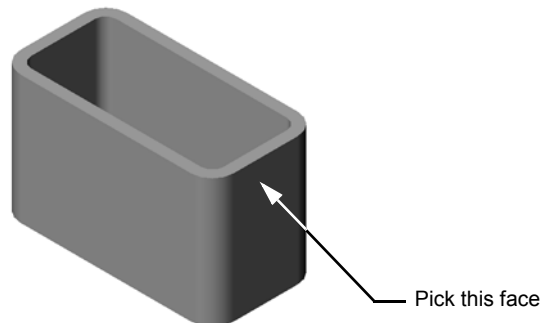
Extruded Cut Feature

The Extruded Cut feature removes material. To make an extruded cut requires a:


- ❑ Sketch plane – In this exercise, the face on the right-hand side of the part.
- ❑ Sketch profile – 2D circle

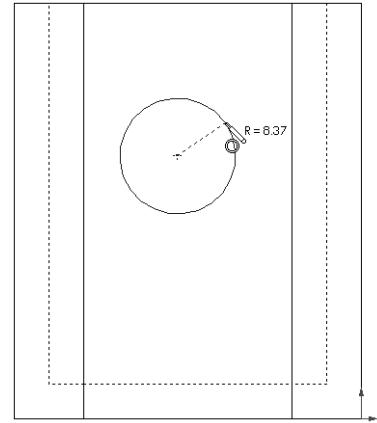
Open a Sketch

- 1 To select the sketch plane, click the right-hand face of the box.
- 2 Click  on the Standard Views toolbar.
The view of the box turns. The selected model face is facing you.
- 3 Open a 2D sketch. Click  on the Sketch toolbar.




Sketch the Circle

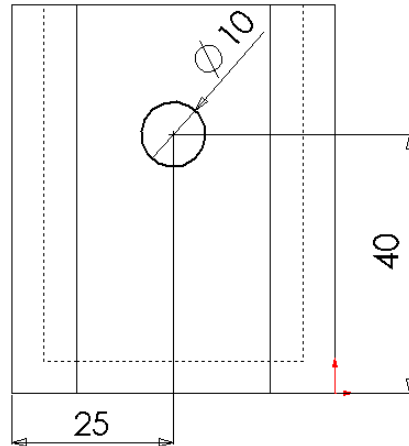
- 1 Click  on the Sketch Tools toolbar.
- 2 Position the pointer where you want the center of the circle. Click the left mouse button.
- 3 Drag the pointer to sketch a circle.
- 4 Click the left mouse button again to complete the circle.





Dimension the Circle

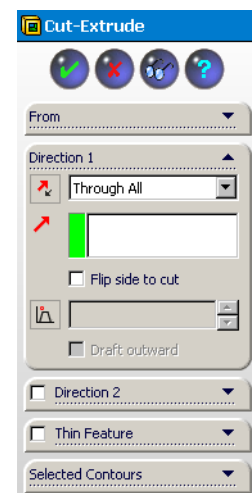
Dimension the circle to determine its size and location.

- 1 Click  on the Sketch Relations toolbar.
- 2 Dimension the diameter. Click on the circumference of the circle. Click a location for the dimension text in the upper right corner. Enter **10**.
- 3 Create a horizontal dimension. Click the circumference of the circle. Click the left most vertical edge. Click a location for the dimension text below the bottom horizontal line. Enter **25**.
- 4 Create a vertical dimension. Click the circumference of the circle. Click the bottom most horizontal edge. Click a location for the dimension text to the right of the sketch. Enter **40**.



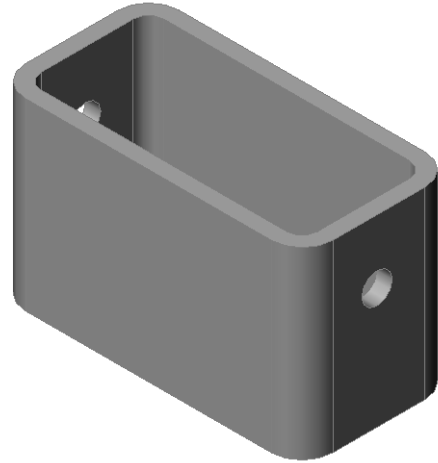
Extrude the Sketch

- 1 Click  on the Features toolbar.
The **Extrude Cut Feature** PropertyManager appears.
- 2 Select **Through All** for the end condition.
- 3 Click .




4 Results.

The cut feature is displayed.




Rotate the View

Rotate the view in the graphics area to display the model from different angles.

- 1 Rotate the part in the graphics area. Press and hold the middle mouse button. Drag the pointer up/down or left/right. The view rotates dynamically.
- 2 Display the Isometric view. Click  on the Standard Views toolbar.

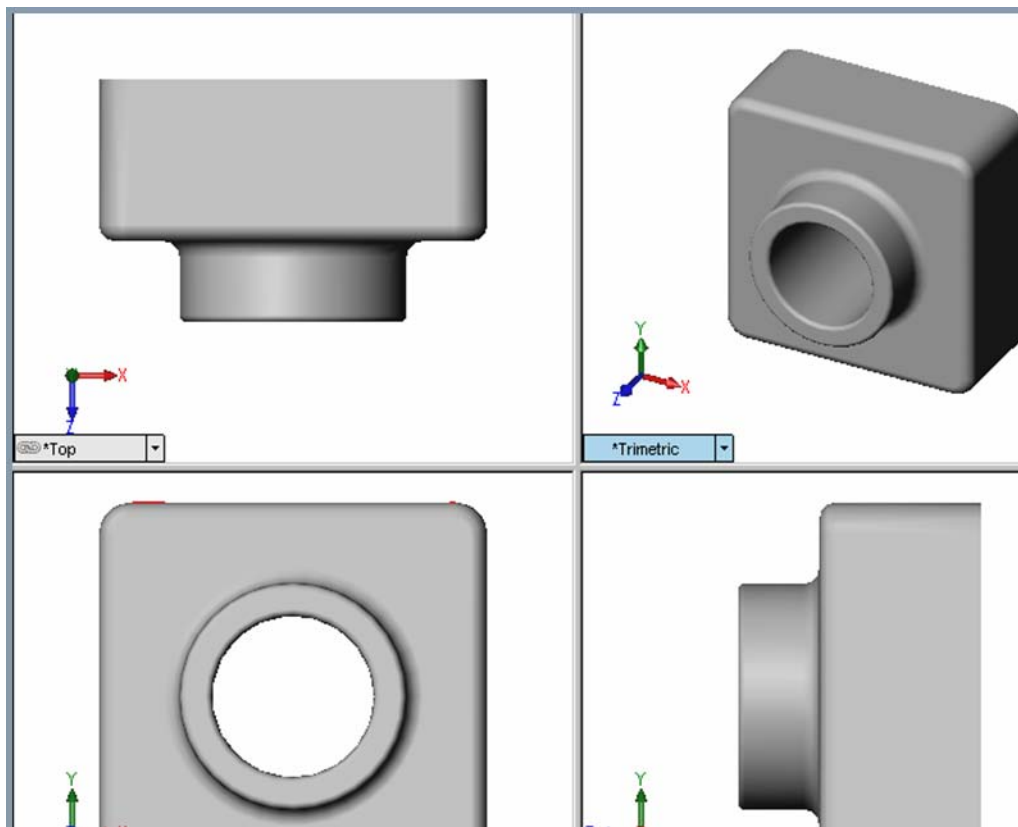
Save the Part

- 1 Click  on the Standard toolbar
- 2 Click **File, Exit** on the Main menu.

Lesson 3: The 40-Minute Running Start

Goals of This Lesson

- ❑ You will be able to create and modify the following part:



Before Beginning This Lesson

- ❑ Complete the previous lesson — Basic Functionality.

Resources for This Lesson

This lesson plan corresponds to *Lesson 1 – Parts* in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See “Online Tutorials” on page v.